



# GNSS Module Application Note (MT3333 series)

Date: 2017-02-24

## Version History

**Title:** GlobalTop GNSS Module Application Notes (MT3333 series)

**Subtitle:** GNSS Module

**Doc Type:** Technical Document

Revision	Date	Editor	Description
A00	2013-01-23	Allen	First Release
A01	2013-08-14	Dylan	Add Gms-g6 module, Gms-b6 module
A02	2013-10-15	Dylan	Add Gmm-g3(B) module
A03	2014-10-28	Rock	Add PA6E-CAM module
A04	2014-11-12	Delano	Revised Gms-g6, Gms-b6 ground plane
A05	2015-07-07	Benson	Add Gmm-3301 module , 2.1.3(b) I2C interface and 2.1.3(c) SPI interface Add 1.8V boost to 3.3V application and chip antenna application
A06	2017-01-23	Yingjie	Add Titan X1( Gms-3302) module Modify the 2.1.2 VBACKUP battery Modify the 2.1.5 Antenna Compliance Design Modify the 2.2 Layout Guideline

## Table of Contents

---

<b>Table of Contents</b> .....	<b>3</b>
<b>Attention</b> .....	<b>4</b>
<b>Technical Support</b> .....	<b>5</b>
<b>1. Introduction</b> .....	<b>6</b>
<b>2. General Rules for Design-in</b> .....	<b>8</b>
<b>2.1 Circuit Design</b> .....	<b>8</b>
2.1.1 Power supply VCC .....	8
2.1.2 VBACKUP backup battery .....	9
2.1.3 UART / I <sup>2</sup> C/ SPI Serial Interface .....	10
2.1.4 GND-Ground .....	11
2.1.5 Antenna Compliance Design .....	12
2.1.6 1PPS .....	18
2.1.7 3D_Fix .....	19
2.1.8 1.8V boost to 3.3V application .....	21
<b>2.2 Layout Guideline</b> .....	<b>23</b>
2.2.1 Layout underneath the GNSS module .....	23
2.2.2 Keep far away from high profile or metal canned components .....	25
2.2.3 Placement .....	27
2.2.4 Trace .....	27
2.2.5 Ground Segmentation .....	28
2.2.6 Ground Plane .....	28
2.2.7 Metal effect on the GNSS module .....	31
<b>3. Thermal Profile for SMD Modules</b> .....	<b>32</b>
<b>4. Troubleshooting</b> .....	<b>34</b>
<b>5. Contact</b> .....	<b>36</b>
<b>Appendix I: Super Capacitor Design</b> .....	<b>37</b>
<b>Appendix II: 50 Ω Antenna Matching</b> .....	<b>38</b>
<b>Appendix III: UART to RS232 Interface</b> .....	<b>39</b>
<b>Appendix IV: UART to USB Interface</b> .....	<b>40</b>
<b>Appendix V: How to efficiently transfer 1PPS through extended distances</b> .....	<b>41</b>
<b>Appendix VI: Caution for Reflow Soldering Process</b> .....	<b>46</b>

## Attention

---

### Please read carefully before you start

- If the GNSS receiver is used inside buildings, tunnels, or besides any huge objects, the GNSS signals might be weakened, cut-off or disturbed. Do not assume the receiver is malfunctioned should the mentioned situation occurs.
- This application note provides the necessary guidelines to successfully design a system using GNSS modules. For detailed module specification, please refer to the corresponding datasheet of GNSS module.
- GNSS Module is an electrostatic-sensitive device, please do not touch GNSS module directly. Please follow ESD safety rules when handling.
- For the first time, it is strongly recommended to test the device outdoors with open sky for at least 10 to 15 minutes to ensure full ephemeris data received.

## Technical Support

---

If you encounter any technical problems or you need certain information yet you do not find it elsewhere in our documents, please feel free to contact us. When writing to us, please follow the form below to help us better serve you:

1. Your company name and website
2. Description of the application and system. Please kindly attach illustrations or figures where applicable.
3. GNSS module type
4. GNSS firmware version
5. Detail descriptions of problem encountered and the conditions by which the problem occurred. Please provide with pictures or videos files where applicable.
  - ✓ Test setup descriptions
  - ✓ The problem or issue addressable in the figure

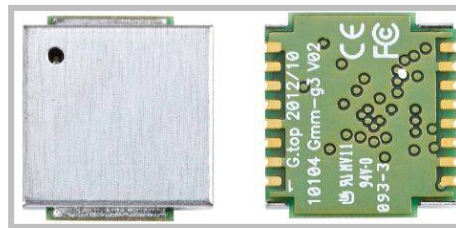
**Technical contact information:** [support@gtop-tech.com](mailto:support@gtop-tech.com)

# 1. Introduction

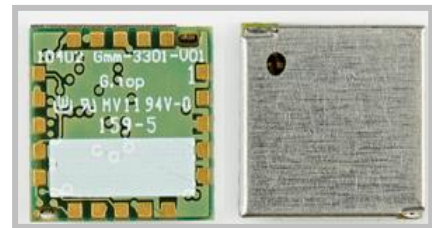
The purpose of this application note is to assist system developer and integrator to get familiar with MT3333 chip based GlobalTop receiver modules during the process of implementation. This document covers the following receiver modules:

GNSS Module Family vs Product Name			
FireFly Series	Product Name	Titan Series	Product Name
FireFly 1	Gmm-g3	Titan1	PA6E-CAM
FireFly 1b	Gmm-g3(B)	Titan 2	Gms-g6
FireFly X1	Gmm-3301	Titan 2B	Gms-b6
		Titan3	Gms-g9
Carrion Series	Product Name	Titan X1	Gms-3302
Carrion 2	Gms-g6a		

**Stand Alone Module**



**FireFly 1 & FireFly 1b (Gmm-g3, Gmm-g3(B))**



**FireFly X1 (Gmm-3301)**

**Built-in Chip Antenna Module**



**Carrion 2 (Gms-g6a)**

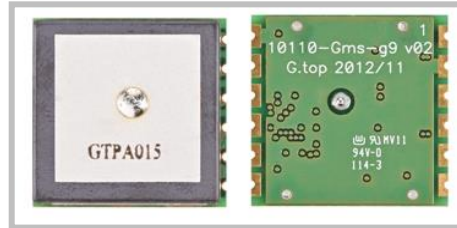
**Built-in Patch  
Antenna  
Module**



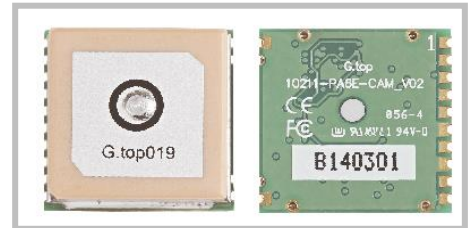
**Titan 2 (Gms-g6)**



**Titan 2B (Gms-b6)**



**Titan 3 (Gms-g9)**



**Titan1 (PA6E-CAM)**



**Titan X1( Gms-3302)**

## 2. General Rules for Design-in

In order to obtain optimal GNSS performances, there are some rules which require attentions for applying the GNSS modules.

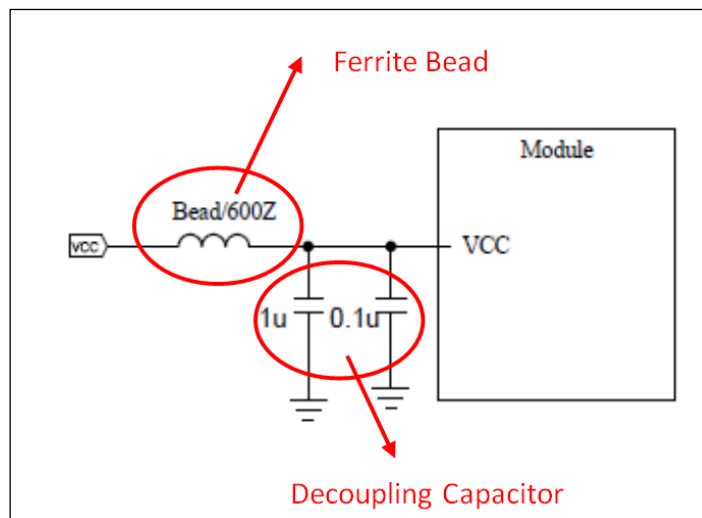
### 2.1 Circuit Design

**Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM, Gmm-3301, Gms-3302**

#### 2.1.1 Power supply VCC

Providing a clean and stable power supply is required for the GNSS module to obtain high-quality performance. Unstable power source will cause a significant negative impact on the performance. To achieve high-quality performance, the VCC ripple must be restricted under 50mV<sub>pp</sub>. Here are some important suggestions for main power circuit design:

- (1) Add ferrite bead, power choke or low pass filter for power noise reduction.
- (2) For power supply, linear regulator (LDO) is better than switch DC/DC power supplier in ripple issue.
- (3) Use suitable decoupling capacitors beside VCC pin for applying stable voltage.



Power design for GNSS module

### 2.1.2 VBACKUP backup battery

**Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, Gmm-3301, Gms-3302**

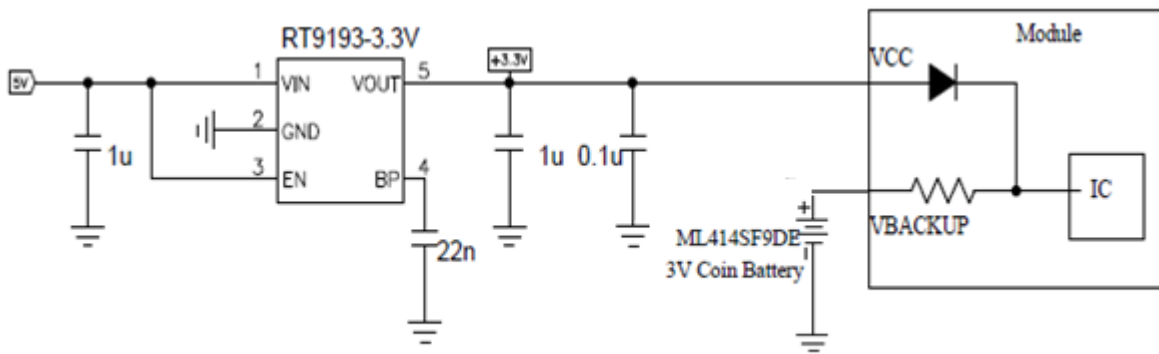
GNSS module has a built-in charging circuit which charges the rechargeable coin battery.

For most system, it is recommended to provide the module with backup power (e.g. Li-Ion rechargeable coin battery, super capacitor).

For “super capacitor reference design”, please refer to Appendix I.

For the backup power, in order to keep RTC operation and retain Ephemeris data in flash memory , which can get a faster TTFF (like short hot start time) and acquire P.V.T(Position, Velocity, Time) information.

If VBACKUP pin don't connect any coin battery, GNSS module will execute cold start, when system is re-start.



Built-in a charging circuit for GNSS module

### 2.1.3 UART / I<sup>2</sup>C/ SPI Serial Interface

#### (a) UART 0 (RX/TX)

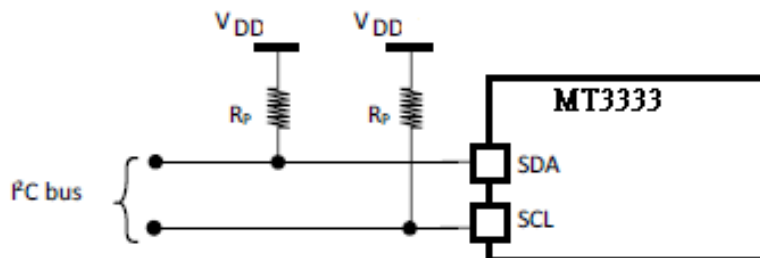
**Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, Gmm-3301, Gms-3302**

- (1) UART is the TTL level interface that carries the baud rate ranging from 4800 bps to 115200 bps.
- (2) Placing damping resistor on the RX and TX of the GNSS module could limit the interference from host MCU or high speed digital logics. Fine tuning the damping resistor is required to efficiently suppress interference. Since the damping resistor is wire wound component, it may function as choke coil.
- (3) Please do not connect diode(s) to RX/TX as it will decrease signal driving capability which might adversely affect RX/TX signal level. In some cases no data output will occur.
- (4) If RS232 logic-level is needed for any particular application, the level shifter will be necessary. For more information, please refer to “UART to RS232 Interface” in Appendix III.
- (5) If USB logic-level is needed for any particular application, please refer to “UART to USB Interface” in Appendix IV for more information.

#### (b) I<sup>2</sup>C (SCL/SDA)

**Compatible Model: Gmm-3301, Gms-3302**

- (1) The I<sup>2</sup>C interface is mainly connected to external devices. MT3333 supports only slave mode (default slave address is 0x10). MT3333 has 256 bytes URAM mode and 8-byte FIFO mode for transmitting and receiving data. The bit rate is up to 400 kb/s for fast mode. In addition, MT3333 supports manual or automatic indicator for data transfer in the slave mode. Device addresses in the slave mode are programmable.
- (2) Pull-up resistor must be added for I<sup>2</sup>C bus:



### (c) SPI (CS/CLK/MISO/MOSI)

#### Compatible Model: Gmm-3301, Gms-3302

- (1) The serial peripheral interface port manages the communication between digital BB and external devices. MT3333 supports only slave mode. The slave has 4 byte-register mode or URAM mode. In the URAM mode, the transmitted and received data size is 256 bytes. The clock phase and clock polarity are selectable. MT3333 supports manual or automatic indicator for data transfer in the slave mode. The bit rate is up to 700kb/s.

#### 2.1.4 GND-Ground

##### Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, Gmm-3301, Gms-3302

Make sure all GND pins of the module are connected to a completed ground plane.

The separation of ground between GNSS module and the rest of the system are recommended to avoid interference. If this setup is not possible, it is best to follow these typical rules: divide to segmentation of ground between digital and analogue system, high current and low current system, and different radiation systems in general (such as GNSS and GPRS)

One way to segment the ground is to place digital and noise component at one corner of the board, while placing analog and quiet components at the opposite corner of the board. Make sure there is no crossing of microstrip or current between the two component sets with ground of each sets to be connected to one point only.

Another way is to place the two different sets at different layers of the board while the ground of each layer is connected to one point only which is preferable located at the border of the board practically.

## 2.1.5 Antenna Compliance Design

### Compatible Models: Gmm-g3, Gmm-g3(B), Gmm-3301, Gms-3302

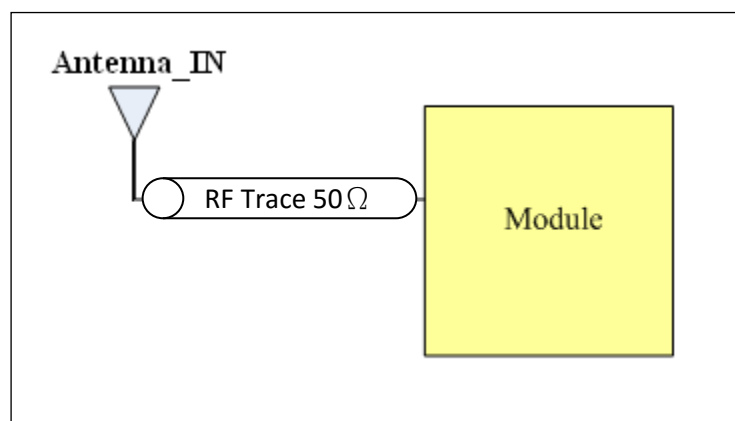
GNSS antenna is a receiving part of the device to acquire weak GNSS signal from sky. Popular solution would be ceramic patch antenna because of its small form factor with low cost. There are two types of antennas: passive and active.

Passive antenna, like patch antenna and chip antenna, is solely by itself without signal amplifier such as LNA. It is required to fine tune for the specific module model to obtain the best signal receiving.

External antenna is a stand alone device, which integrates with LNA and patch antenna with a RF cable for higher gain. It perform better than passive antenna.

### Designing Patch antenna with GNSS module

- (1) In general, a 50 $\Omega$  patch antenna will work well with GNSS module. The antenna can be connected to the Antenna IN pin with a 50 $\Omega$  impedance trace.
- (2) Please keep the patch antenna far away from noise source such as switching power supply, high speed digital logics or radar wave guide.
- (3) The 50 $\Omega$  trace should be kept as short as possible to reduce the chance of picking up the noise from air and PCB. A simple direct-line trace is most recommended.
- (4) If needed, a matching circuit could be placed between patch antenna and GNSS module. The matching circuit design should be discussed with the module and patch antenna maker.
- (5) For 50  $\Omega$  matching, please refer to “50  $\Omega$  Antenna Matching” in Appendix II.



PCB trace design for antenna impedance matching

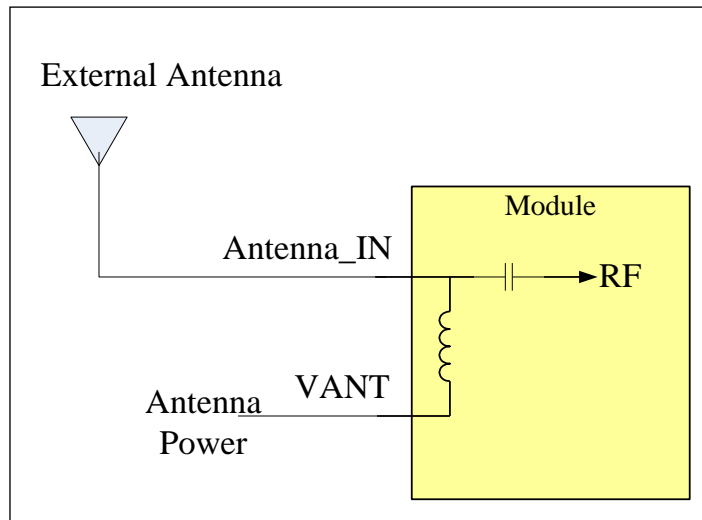
**Select External Antennal architecture with GNSS module**

An external antenna requires DC power to function. A typical method is to feed DC power into the RF trace. The external antenna can extract the power from the RF trace; thus we can use the RF trace to send RF signal and DC voltage. To fully achieve delivery, we use a RF chock coil to couple the DC power to RF trace.

In general, GlobalTop modules provide the circuit connecting external antenna and power feeding method as depicted in the figure below. Please select the module prudently while applying the design with an external antenna.

**Mode 1(i.e. Gmm-g3, Gmm-g3(B)) :**

The power supply for the external antenna needs to be externally fed into the module via the VANT pin and connect antenna to Antenna IN pin.

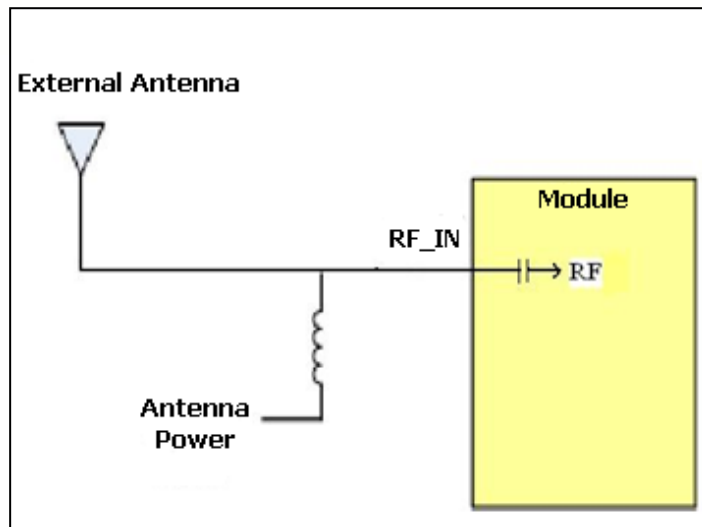


Mode 1: (Internal chock coil)

**Mode 2 (i.e. Gmm-3301):**

The power supply needs to be externally provided and it is connected directly to the External Antenna via chock coil.

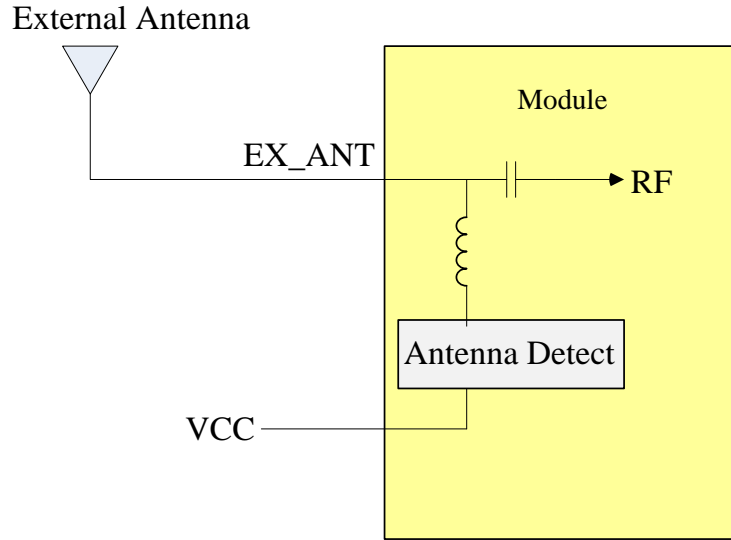
Note: Chock Coil for reference: LQG15HS33NJ02D (Murata)



Mode 2: (External chock coil)

**Mode 3 (i.e. PA6E-CAM, Gms-3302) :**

The power supply for the external antenna needs to be externally fed into the module from the VCC pin directly and connect antenna to EX\_ANT pin.



Mode 3: (Internal chock coil)

Mode \ Module	Gmm-g3, Gmm-g3(B)	Gmm-3301	PA6E-CAM Gms-3302
Mode 1 (via the VANT) (Internal chock coil)	✓		
Mode 2 (External chock coil)		✓	
Mode 3 (from the VCC) (Internal chock coil)			✓

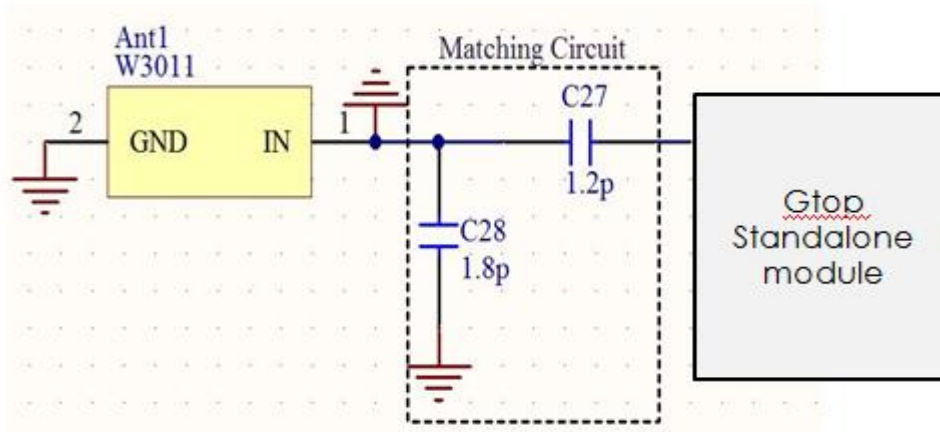
**Design chip antenna with GNSS module**

Different vendors have different reference design, so we list two chip antenna sources. One is Pulse and the other is Unictron.

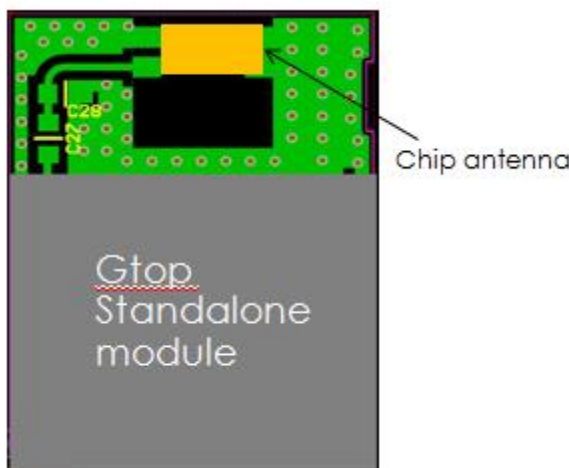
- a. Antenna source: Pulse

Schematic design

Antenna source : Pulse



Pcb layout



When you select the Antenna source, you can check its datasheet first, and reserve tuning RF match component's footprint (like:C27,C28).Customers can base on its pcb size and housing to tune optimal value and meet GNSS's frequency, then can get good reception effect.

Pulse web site:

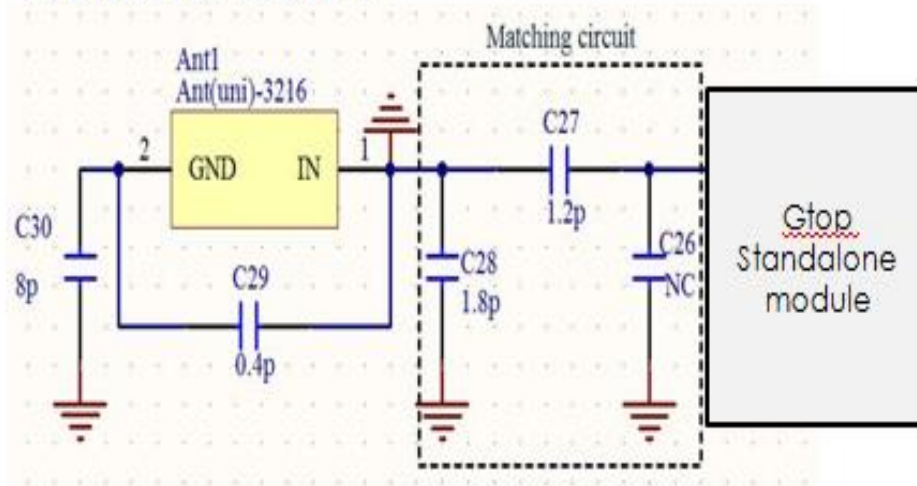
[www.pulseeng.com/antennas](http://www.pulseeng.com/antennas)

**Note: C27,C28's value are still based on your actual trace to tune it.**

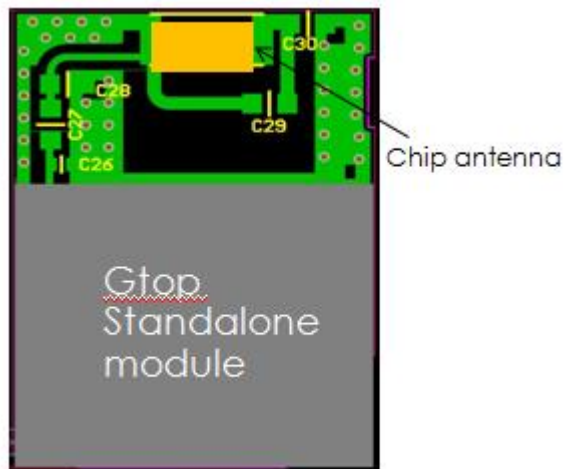
b. Antenna source: Unictron

Schematic design

Antenna source : Unictron



pcb layout



When you select this source, you can check its datasheet first, and reserve tuning RF match component's footprint (like:C26,C27,C28,C29,C30) .Customers can base on its pcb size and housing to tune optimal value and meet GNSS's frequency, then can get good reception effect.

Unictron web site:

<http://www.unictron.com/index/>

**Note: C26,C27,C28,C29,C30's value are still based on your actual trace to tune it.**

## External Antenna of Specification

When select your external antenna, refer to the specification shown in the table below.

For Gmm-g3 module application, below is GPS/GLONASS External Antenna

Characteristic	Specification
Polarization	Right-hand circular polarized
Frequency Received	1.575GHz~1.615GHz
Power Supply	3.3V
DC Current	5mA<IDC<11mA at 3.3V
Total Gain	27±3dB
Output VSWR	2.0
Impedence	50Ω
Noise Figure	2dB

Table 1

For Gmm-g3(B) module application, below Beidou/GPS External Antenna.

Characteristic	Specification
Polarization	Right-hand circular polarized
Frequency Received	1.561GHz~1.575GHz
Power Supply	3.3V
DC Current	3mA<IDC<30mA at 3.3V
Total Gain	27±1dB
Output VSWR	2.0
Impedence	50Ω
Noise Figure	1.5dB

Table 2

### 2.1.6 1PPS

**Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, Gmm-3301, Gms-3302**

1PPS signal is an output pulse signal used for timing application. Its electrical characteristics are:

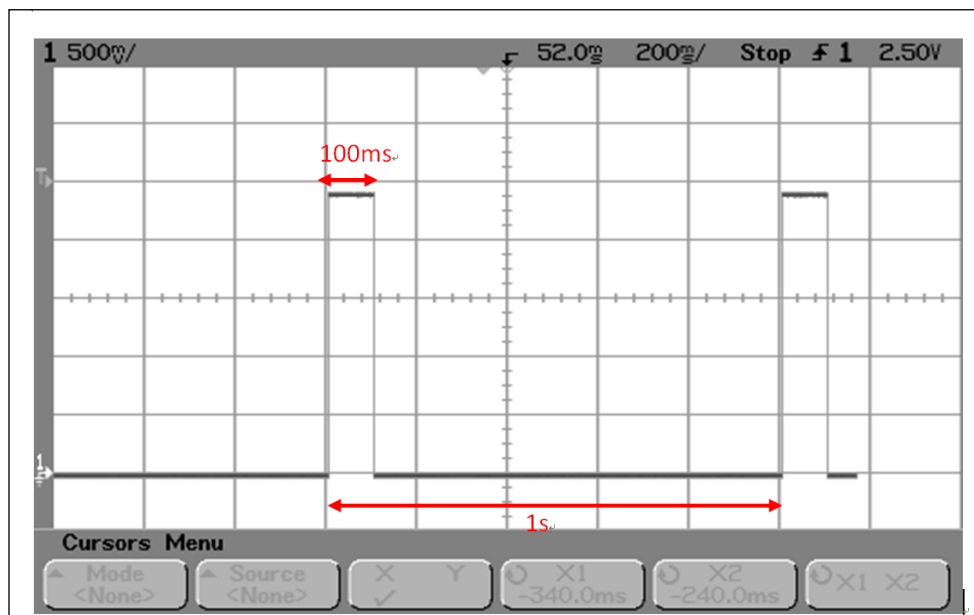
Low Voltage level: 0~0.4V

High Voltage level: 2.8~3.1V

Period: 1s

Accuracy (jitter): $\pm 10$ ns

100ms pulse width duration (Firmware customization for pulse width duration is available)



1PPS signal and its pulse width with 100ms duration

#### Free run 1PPS output before 3D\_FIX

GlobalTop standard GNSS module outputs 1PPS signal after the module obtained 3D\_FIX. This is a factory default setting.

For an always 1PPS signal output configuration, regardless of 3D\_Fix or not, this is available to our customer via firmware customization service. Please be noted that the accuracy before 3D\_Fix on the 1PPS is a free run and will lock until GPS 3D\_Fix.

#### Cable delay compensation

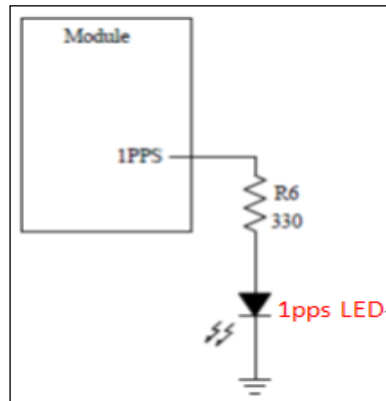
In some cases a long distance connection (~300m) may be needed. For a timing application, the cable length is critical.

For more information on 1PPS signal transmission delay please refer to Appendix V.

Delay caused by the cable length could be compensated with firmware setting. Please contact us if that is useful for your implementation.

### LED indicator for 1PPS signal

For 1PPS LED indication, user may connect a LED indicator with a 330ohm resistor in series.



1PPS signal design for IO

### 2.1.7 3D\_Fix

**Compatible Model: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6**

3D\_Fix signal is an output pulse signal used for GNSS fix application. Its electrical characteristics are:

Low Voltage level: 0~0.4V

High Voltage level: 2.8~3.1V

Source current (Max.): 14mA, sink current 14mA

#### Signal output before 2D Fix:

Duration: 1 second high and 1 second low.

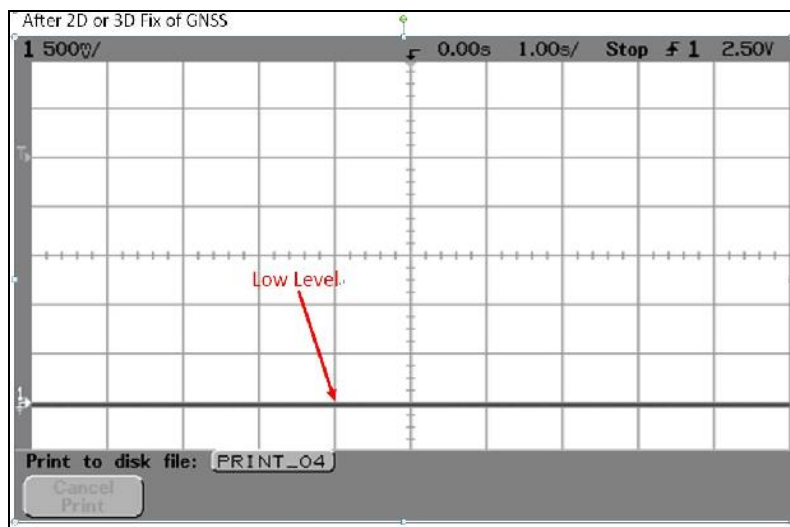
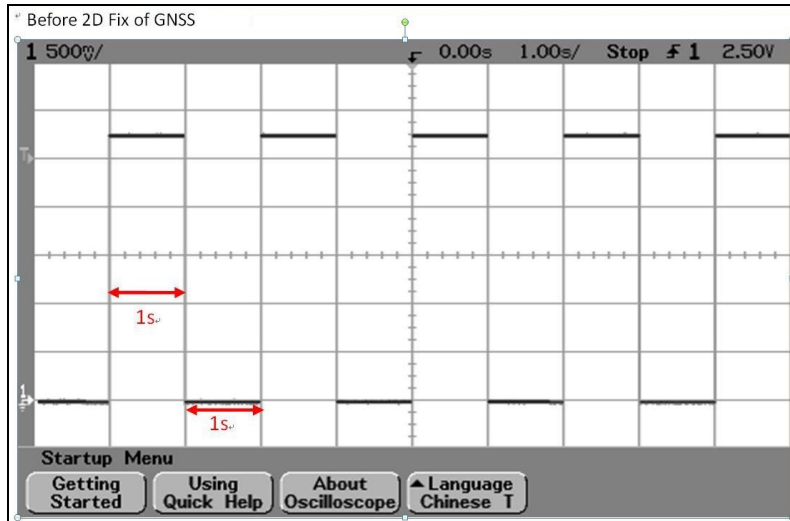
Period: 2s

(Firmware customizable)

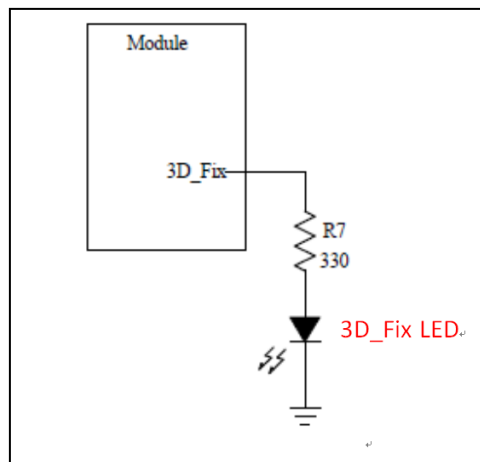
#### Signal output after 2D or 3D Fix:

Duration: Constant low level output

(Firmware customizable)

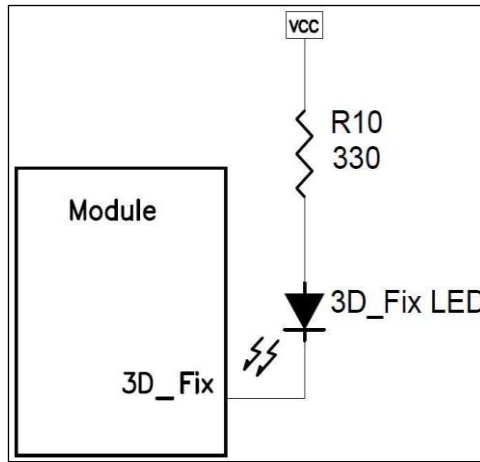


For 3D\_Fix indication, user may connect a LED with a 330ohm resister in series in two ways showing in the two figures below.



3D\_Fix signal design for IO

(LED ON when high, LED OFF when low)



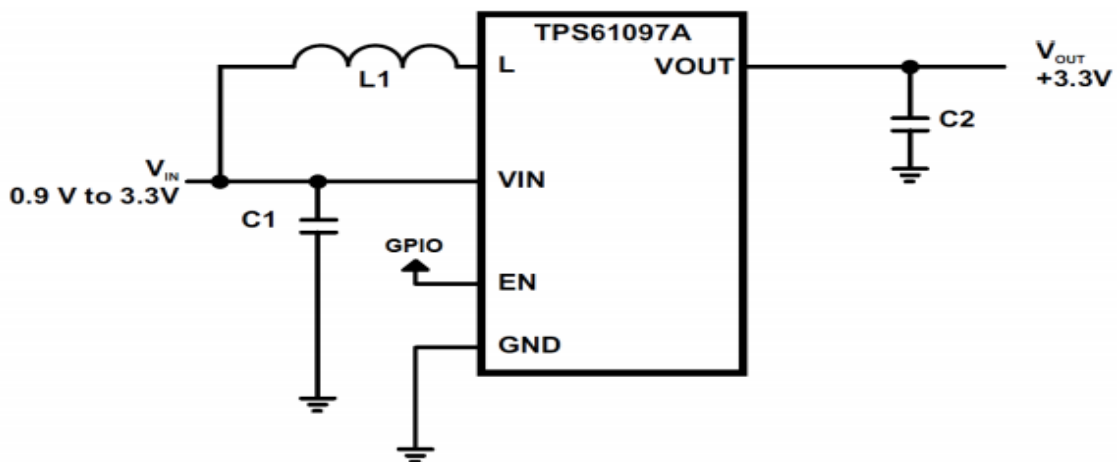
3D\_Fix signal design for IO

(LED ON when low, LED OFF when high)

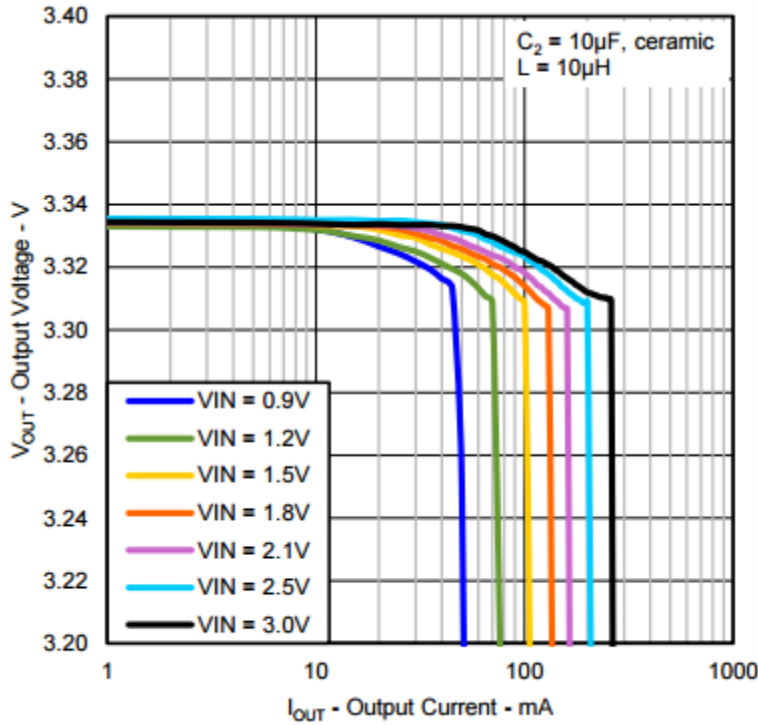
### 2.1.8 1.8V boost to 3.3V application

In the 1.8V Voltage ' system, if you want to use 3.3V's GNSS module, we need to consider two conditions, first one is power supply translation, second is signal level shift; About power supply translation, we can adopt boost circuit which can let 1.8V boost to 3.3V (refer to below figure), TPS61097A-33 is Ti's boost IC which can support boost function, in its application, C1 and C2 need to use 10uF and L1 is 10uH, it can support about 100mA output; about capacitor and inductor's placement can refer it's application note , you can go to Ti web site:

<http://www.ti.com/lit/ds/symlink/tps61097a-33.pdf>

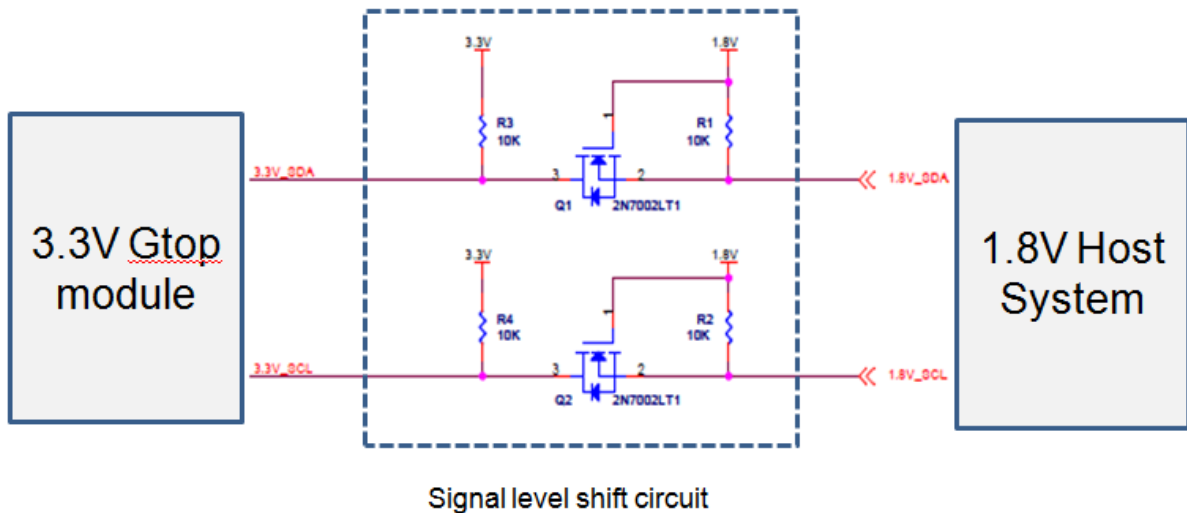


(Application schematic)



(Output Voltage vs Output Current)

About signal level shift, when user's host system is 1.8 Voltage and Gtop module is 3.3 Voltage, so host system can control Gtop module by adopting signal level shift circuit (refer to below), R1~R4's values are default value, In actual design, user can adjust their values to achieve control; 2N7002L can select low RDS(on), it will be reduced power consumption and voltage's drop.



## 2.2 Layout Guideline

Please follow the layout guideline for the right GNSS module when design.

### 2.2.1 Layout underneath the GNSS module

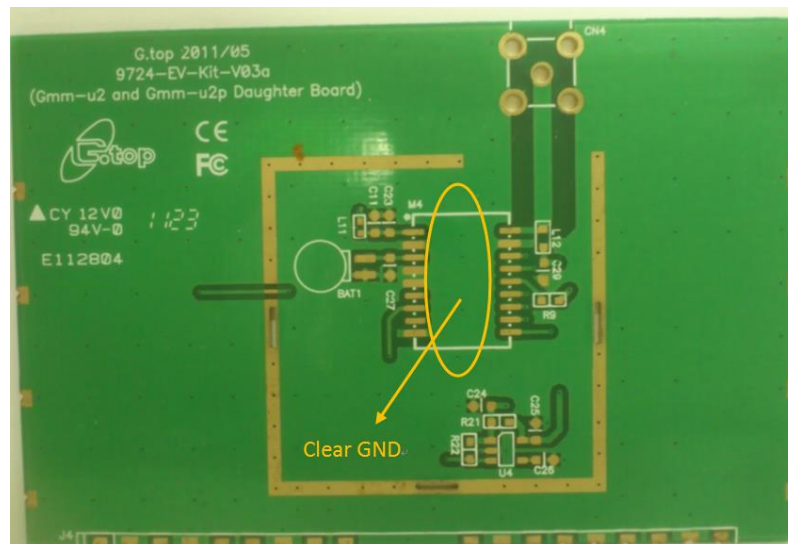
**Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, Gmm-3301, Gms-3302**

In general, GNSS modules have high receiving sensitivity at around -165dBm. During the hardware integration, try to avoid noise or harmonic in these bands to prevent unnecessary reception degradation (Beidou 1561.098MHz ±2.046MHz and GPS 1572.42MHz±2MHz or GLONASS 1598.0625~1605.375MHz).

In modern GNSS positioning end products which include a lot components such as LCD screen, MCU, high speed digital signal(access memory card) and RF system (i.e. Cellular module, BT, Wi-Fi, DVB-T). Please far away these components to GNSS module, In order to avoid the noise interference, otherwise it will cause poor GNSS RF reception ability.

#### For modules Gmm-g3, Gmm-g3(B) & Gmm-3301

Don't leave any trace or mark underneath the GNSS module where the circled area is pointed at the figure below. In other words, give GNSS module a clean GND plane.



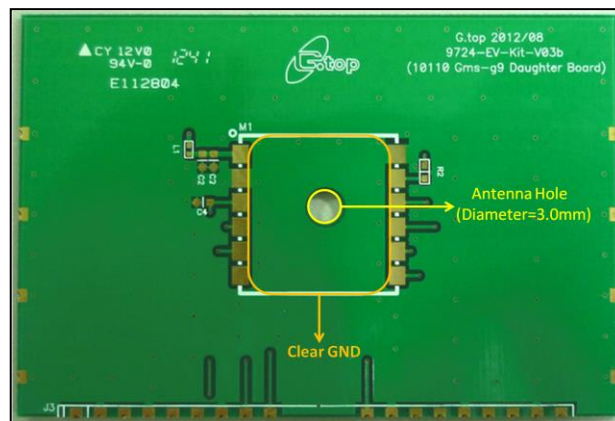
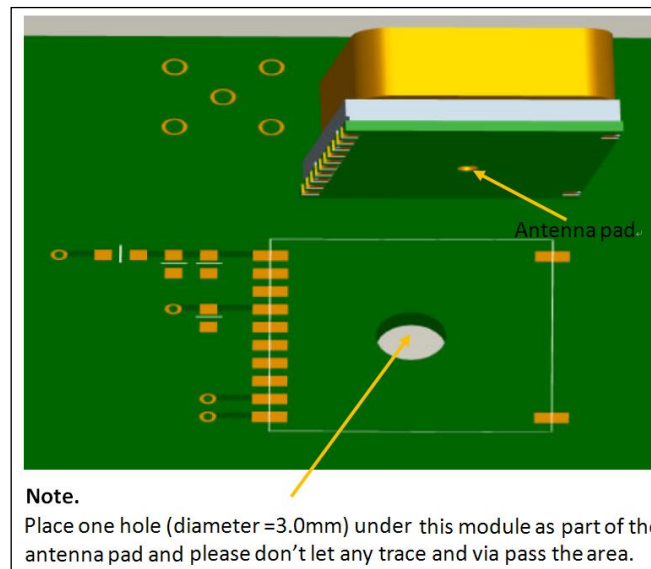
GNSS module on a clean GND plane

✗ Don't place any trace such as I<sup>2</sup>C (SCL/SDA), SPI (CLK/MISO/MOSI), UART (TX/RX) underneath the GNSS module, otherwise it will caused sensitivity decrease.

**For modules Gms-g9, Gms-g6, Gms-b6 & PA6E-CAM**

Place a hole according to the location of RF feeding pin in module. The hole will provide an area to fit the RF feeding pin on your system PCB and provide the appropriate the size of hole.

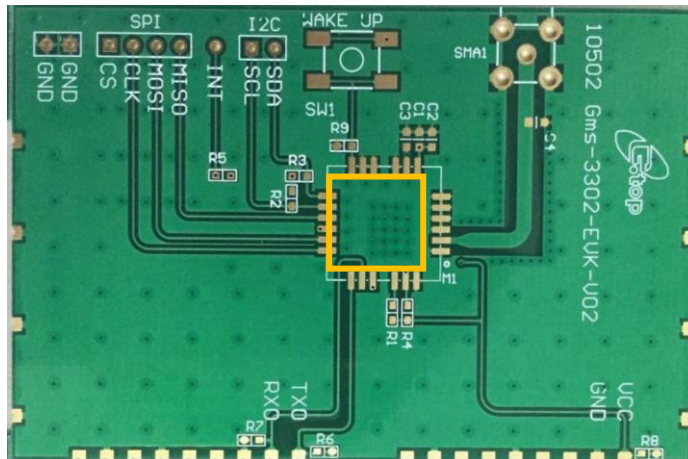
※Don't place any high speed trace such as **I<sup>2</sup>C (SCL/SDA), SPI (CLK/MISO/MOSI), UART (TX/RX)** underneath the GNSS module, otherwise it will cause sensitivity decrease.



### For modules Gms-3302

In order to avoid the interference, please place many vias on the two sides of RF trace which is from module to SMA/RF connector on your system PCB. (refer to below figure)

※ Don't place any high speed trace such as **I<sup>2</sup>C (SCL/SDA)**, **SPI (CLK/MISO/MOSI)**, **UART (TX/RX)** underneath the GNSS module, otherwise it will cause sensitivity decrease.

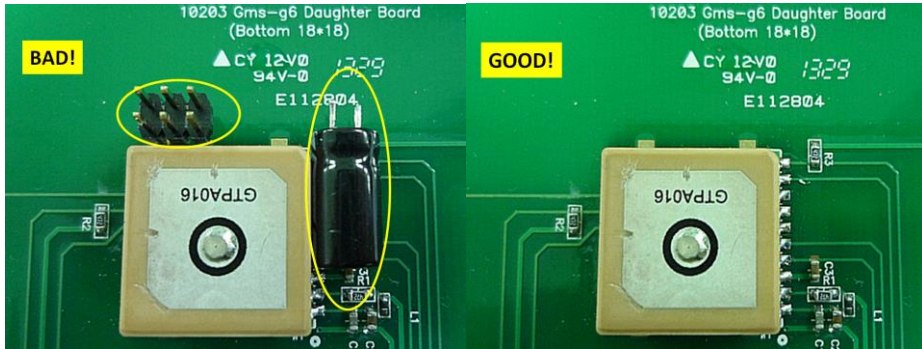
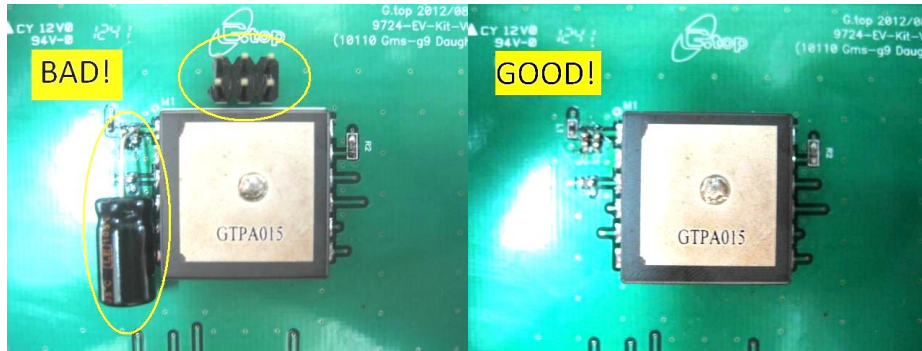


### 2.2.2 Keep far away from high profile or metal canned components

**Compatible Model: Gms-g6a, Gms-g9, Gms-g6, Gms-b6, PA6E-CAM, Gms-3302**

It is good practice to place GNSS module far away from any high profile component especially with metal case enclosed components such as E-CAP, Coin battery, Pin Header. The Antenna field pattern can be affected and pattern distortion can be happened. At the worst case it will decrease the GNSS signal up to ~10dB.





### 2.2.3 Placement

**Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM, Gmm-3301,Gms-3302**

Place the decoupling capacitors for VCC close to GNSS module.  
Place the damping resistors for TX/RX close to GNSS module.

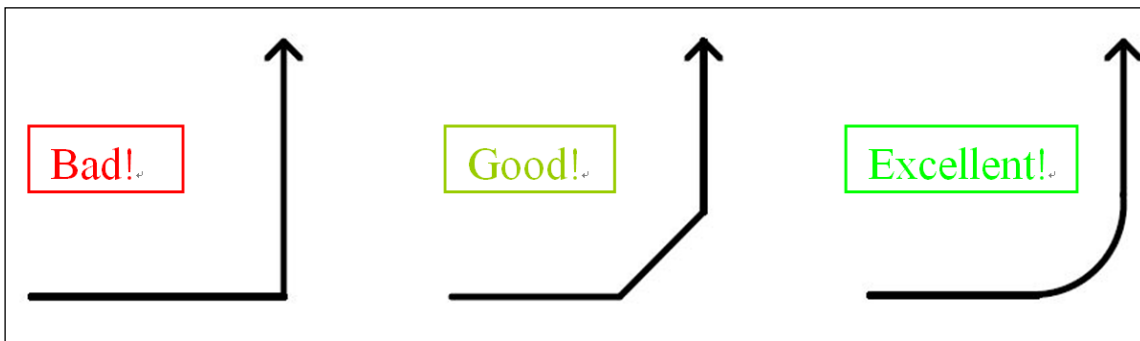
#### Do not place:

- GNSS module in proximity to high-speed digital processing circuitry.
- GNSS module in proximity to high-current switching power circuitry.
- GNSS module in proximity to clock sources circuitry.

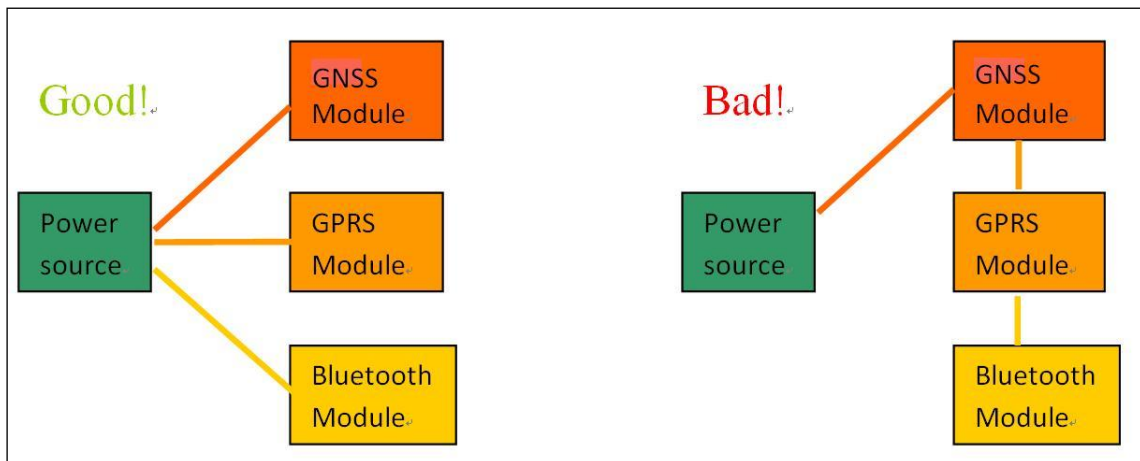
### 2.2.4 Trace

**Compatible Model: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM, Gmm-3301,Gms-3302**

- (1) The USB differential signals should be trace closely and of equal length for better noise immunity and minimum radiation.
- (2) Apply 50 ohm impedance RF trace for correct impedance matching.
- (3) Any right angle turn in a trace routing should be done with two 135 degree turn or an arc turn.



It is better to have independent trace of power source for any device.



## 2.2.5 Ground Segmentation

**Compatible Models: Gmm-g3, Gmm-g3(B), Gms-g6a, Gms-g9, Gms-g6, Gms-b6,PA6E-CAM, Gmm-3301,Gms-3302**

In general, the separation of ground between GNSS module and the rest of the system is recommended to avoid interference. If this is not possible, it is best to follow these typical rules: segmentation of ground between digital and analogue system, high current and low current system, and different radiation systems such as GNSS and GPRS.

One way to segment the ground is to place digital and noise component at one corner of the board, while placing analog and quiet components at the opposite corner of the board. Make sure there is no crossing of microstrip or current between the two component sets with ground of each sets to be contacted to one point only.

Another way to do this is the place the two different sets at different layers of the board, while the ground of each layer is contacted in one point only which is preferable located at the border of the board practically.

## 2.2.6 Ground Plane

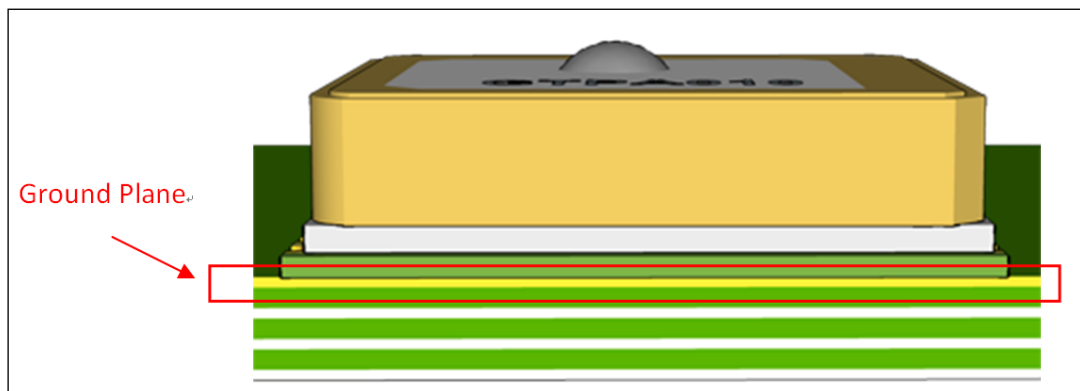
**(1) Compatible Models: Gms-g9,PA6E-CAM, Gms-g6, Gms-b6,Gms-3302**

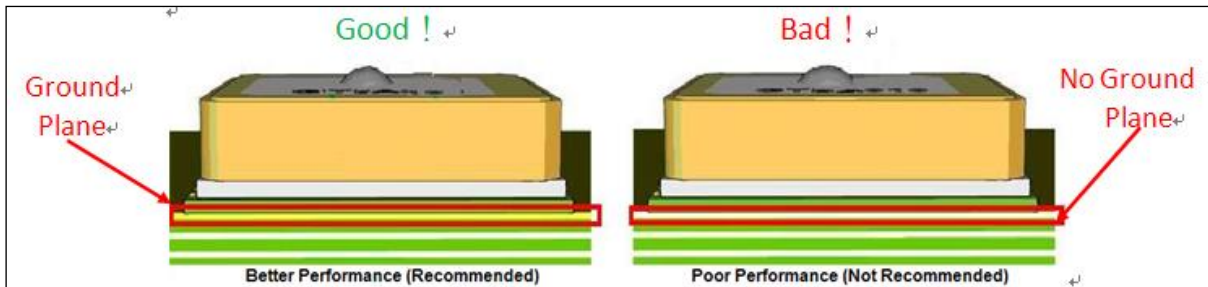
**For Gms-g9, PA6E-CAM, Gms-g6, Gms-b6,Gms-3302** A large GND plane directly underneath the module could enhance the magnetic-field line of the antenna for better GNSS signal reception. Typically it will improve ~2dB maximum.

It is strongly recommended to have a ground plane designed underneath the GNSS module as big as possible.

The recommended thickness for the ground layer is 0.5 to 1 OZ (0.0175 to 0.035 mm)

It is best to place the ground plane on the top layer of the PCB, directly underneath the GNSS module as shown in the figure below.



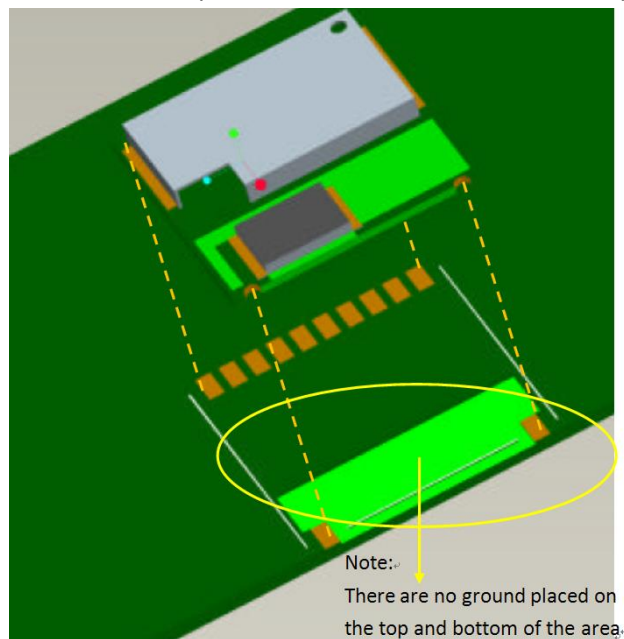


Ground Plane for FR4 Design

**(2)Compatible Model: Gms-g6a**

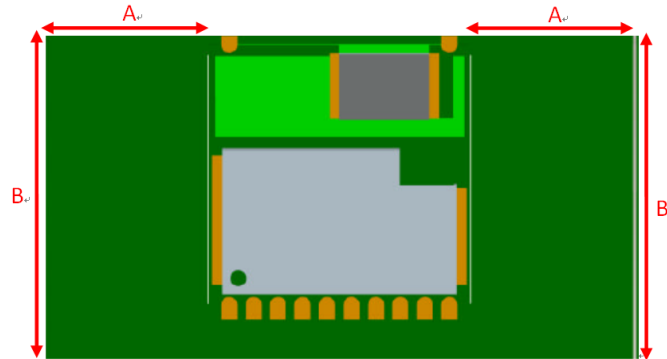
**For Gms-g6a** A built-in Chip Antenna module, a special ground plane is required to improve chip antenna reception performance.

As shown in the figure, the area circled where the chip antenna area is when placed on the PCB, no ground plan should be placed on this chip antenna area and also all other layers underneath.



Note: The clearance area size at least 15x6.3mm.

To enhance the antenna efficiency for better GNSS signal reception, it is recommended to cover the rest of the PCB top layer with ground plane except where the chip antenna seats as shown below.



Variation in reception sensitivity according to the different size of the GND plane

### 2.2.7 Metal effect on the GNSS module

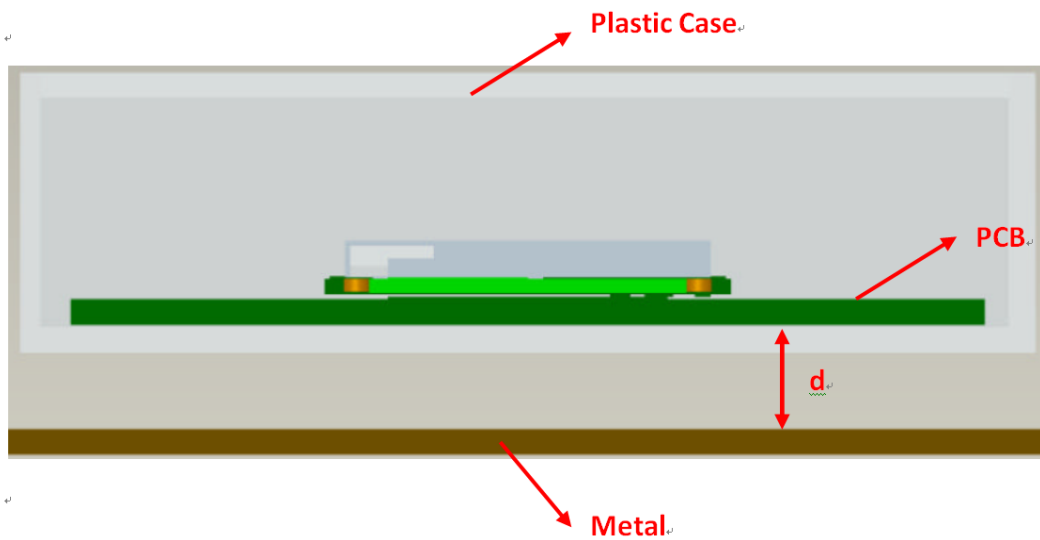
#### Compatible Model: Gms-g6a

For Gms-g6a Chip Antenna is a sensitive component with environment.

A consideration regarding to the housing and the environment where it was placed should be carefully taken.

- (1) The housing (case) of the host device should not be made of or include any conductive material for the basic antenna reception.
- (2) The host device should not be placed underneath of a conductive material such as metal case, metal film or plastic housing coated in conductive material, which blocks GNSS signal from reaching the antenna; and if there is metal material placed underneath the chip antenna, this will also decrease the antenna reception performance.

As shown in the figure, the distance (d) between metal and module should be kept as far as possible.



The table would give an insight on how reception performance can be negatively affected in regard to the distance between the metal and the mounted module.

Distance (mm)	Sensitivity decrease in dB
0mm	-14 dB ~-18dB
1mm	-7dB~-9dB
3mm	-6dB ~-7dB
5mm	-3dB~-5dB
7 mm	-2dB~-4dB
10mm*	0dB

### 3. Thermal Profile for SMD Modules

The following information is Pb-Free compliant.

The details are for reference only.

#### SMT Reflow Soldering Temperature Profile

Average ramp-up rate (25 ~ 150°C): 3°C/sec. max.

Average ramp-up rate (270°C to peak): 3°C/sec. max.

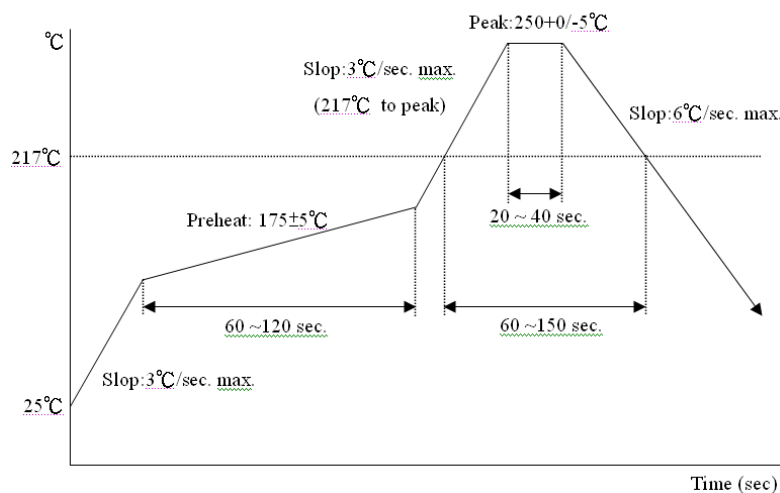
Preheat: 175 ± 25°C, 60 ~ 120 seconds

Temperature maintained above 217°C: 60~150 seconds

Peak temperature: 250 +0/-5°C, 20~40 seconds

Ramp-down rate: 6°C/sec. max.

Time 25°C to peak temperature: 8 minutes max.



## SMT Solder Mask

Please use the dimension of PCB pad as reference and shrink the size by 0.1 to 0.2 mm and use that as layout for paste mask. (For PCB pad layout, please see “Recommended PCB pad layout” on individual GNSS module data sheet)

## Manual Soldering

Soldering iron: Bit Temperature: Under 380°C | Time: Under 3 second.

## 4. Troubleshooting

### How to check for the working status of GNSS module?

The first thing to check for is the NMEA sentence output through Tx using various application tools. For example: you may use windows default tool –Hyperterminal in WinXP or you may use other GNSS application program.

If there is no NMEA output at the TX pin, this indicates that the module is currently not working. Please double check your schematic design. The list down below is a go through list of possible items to check.

#### Item 1: VCC

The voltage should be kept between 3V to 4.3V. **(Typical: 3.3V). Please double-check.**

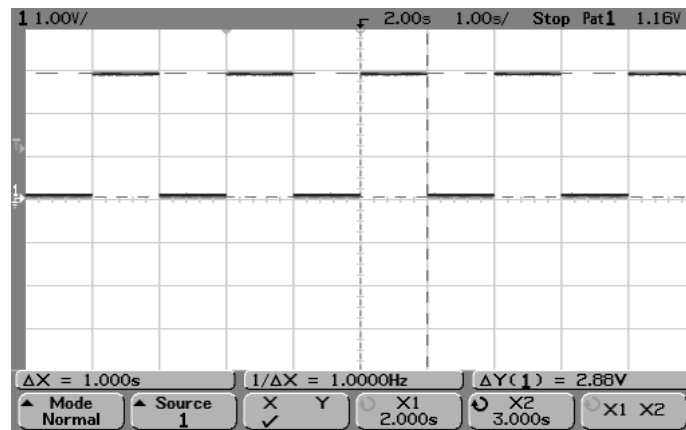
#### Item 2: VBACKUP

The voltage should be kept between 2.0V~4.3V. **(Typical: 3.0V)**. If backup battery is not installed for the VBACKUP pin, the pin could still be measured as the voltage comes from the built-in battery recharging circuit. It is recommended to provide power to VBACUP pin as it is used to keep RTC time running and stored navigation data. **Please double-check**

#### Item 3: 3D-FIX

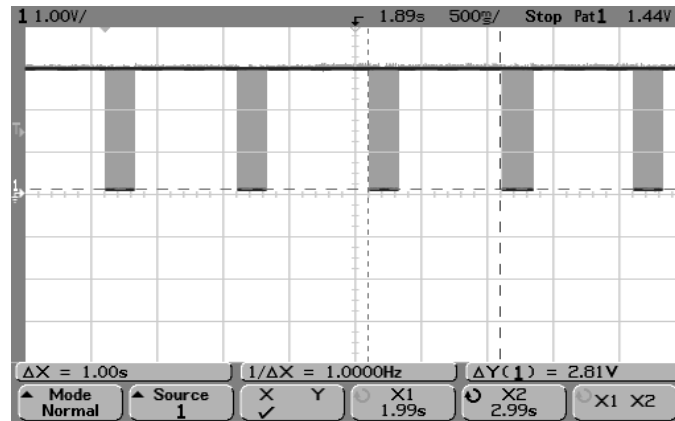
If all the measurements are within the specifications, please also measure 3D-FIX signal. Before 2D Fix, **the pin should output 1 second high followed by 1 second low signal.**

**Notes:** This is not applicable to those with customized 1PPS which has an always 1PPS signal configuration as described in “Free run 1PPS” in **section 2.1.6 1PPS**.



**Item 4: TX0**

The UART transmitter of the module, it outputs the GNSS NMEA information for application.



## 5. Contact

---

For help or further information, please contact us via the following methods:

### GlobalTop Technology Inc.

#### Tainan Science-Based Industry Park Headquarter:

**Address:** No.16 Nan-ke 9th Road Science-based Industrial Park, Tainan 74147, Taiwan

**Tel:** +886-6-505-1268

**Fax:** +886-6-505-3381

**Sales Email:** [sales@gtop-tech.com](mailto:sales@gtop-tech.com)

**Technical Support Email:** [support@gtop-tech.com](mailto:support@gtop-tech.com)

## Appendix I: Super Capacitor Design

### About Super Capacitors

Super capacitor has lifetime similar to that of aluminum electrolytic capacitor. The service life of a super capacitor is greatly dependent on the operating temperature, humidity, applied voltage, current and backup time. Therefore, the service life is determined based on the backup time set by the customer.

### How to calculate the backup time

The example below teaches how to calculate the backup time.

**$T = C (V_0 - V_1) / (I + I_L)$**

Capacitor's voltage

**Example:**

$V_0 = 2.5V, V_1 = 1.8V, I = 10\mu A, C = 0.2F$

$T = 0.2 ( 2.5 - 1.8 ) / ( 10 \text{ e-}6 + 0.2 \times 10^{-6} )$

$= 0.14 / 10.2 \times 10^{-6}$

$= 13,725 \text{ seconds}$

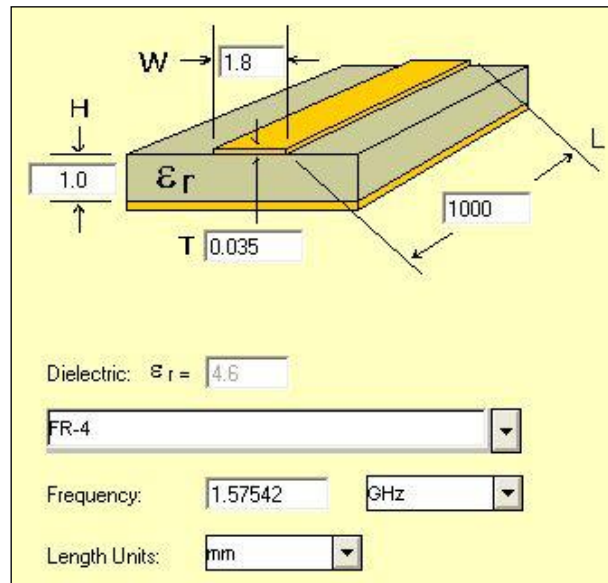
$= 3.8 \text{ hours}$

T: Backup time (second)  
 C: Capacitance of Gold Capacitor (F)  
 V<sub>0</sub>: Applied voltage(V)  
 V<sub>1</sub>: Lower limit voltage for memory backup(V)  
 I: Current during backup(A)  
 I<sub>L</sub>: Leakage current(A) about 0.2uA

## Appendix II: 50 Ω Antenna Matching

We used AppCAD tool to simulate 50 Ω impedance for RF PCB layout.

RF line width (W)	PCB FR4 Thickness (H)	Dielectric parameter	Copper Thickness an ounce
1.8mm	1mm	4.6	0.035mm



Note:

For multiple-layer layout, designer could place a ground layer in the 2nd layer to minimize the trace width in specific PCB (such FR4) and impedance.

For impedance calculation, free software to calculate trace width or impedance is available.

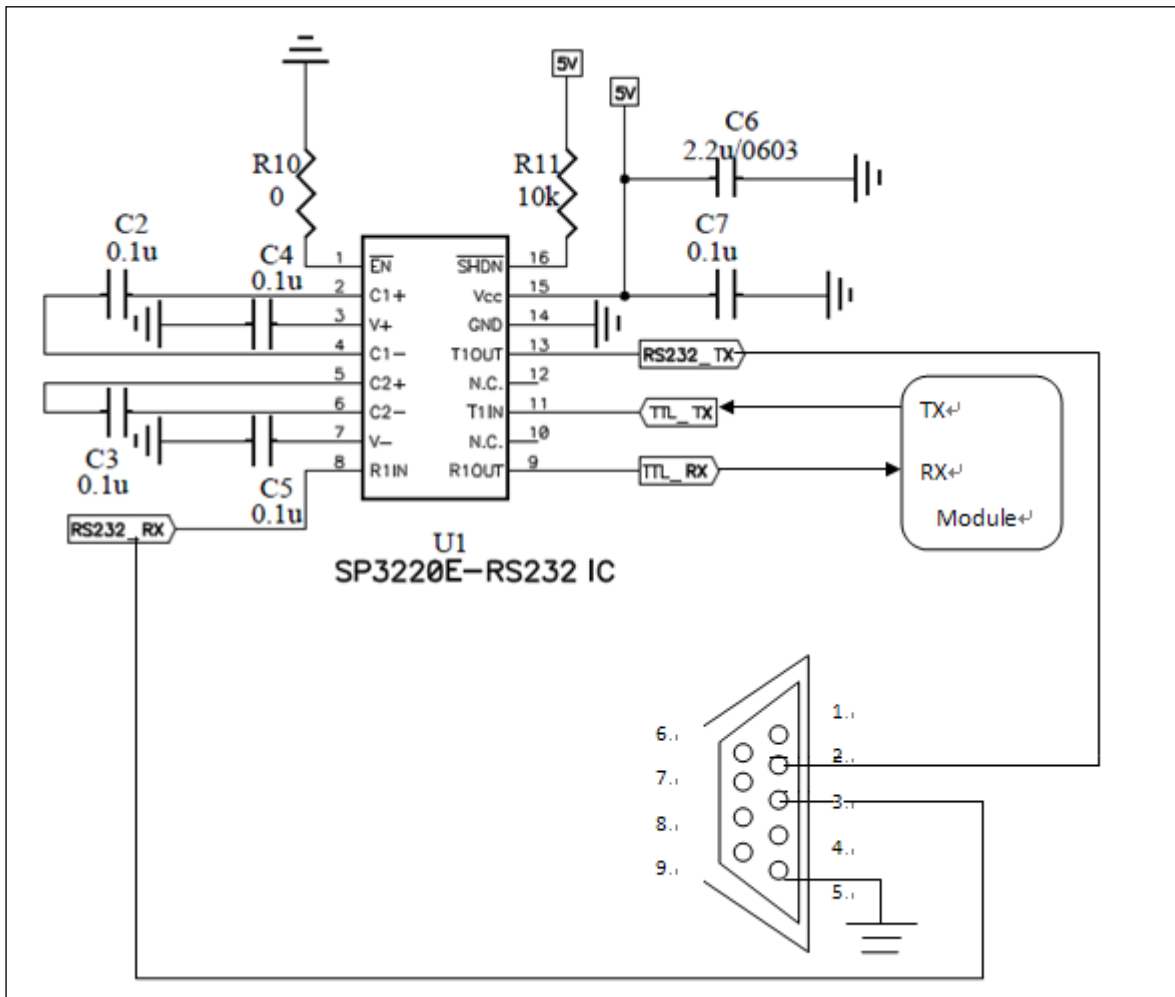
Please refer to the website for one such software:

<http://web.awrcorp.com/Usa/Products/Optional-Products/TX-Line/>

## Appendix III: UART to RS232 Interface

Typically RS232 or USB interface is required to connect the PC to the GlobalTop GNSS module for communication. Majority of GlobalTop modules use a set of communication port in TTL-logic. Some newer ones supports for direct USB connection. A bridge IC is needed for RS232 signal conversion. The supported baud-rates are: 4800, 9600, 14400, 19200, 38400, 57600, and 115200 bps.

Please refer to the reference circuit below for RS232 signal conversion. SP3220 IC is used here as an example.

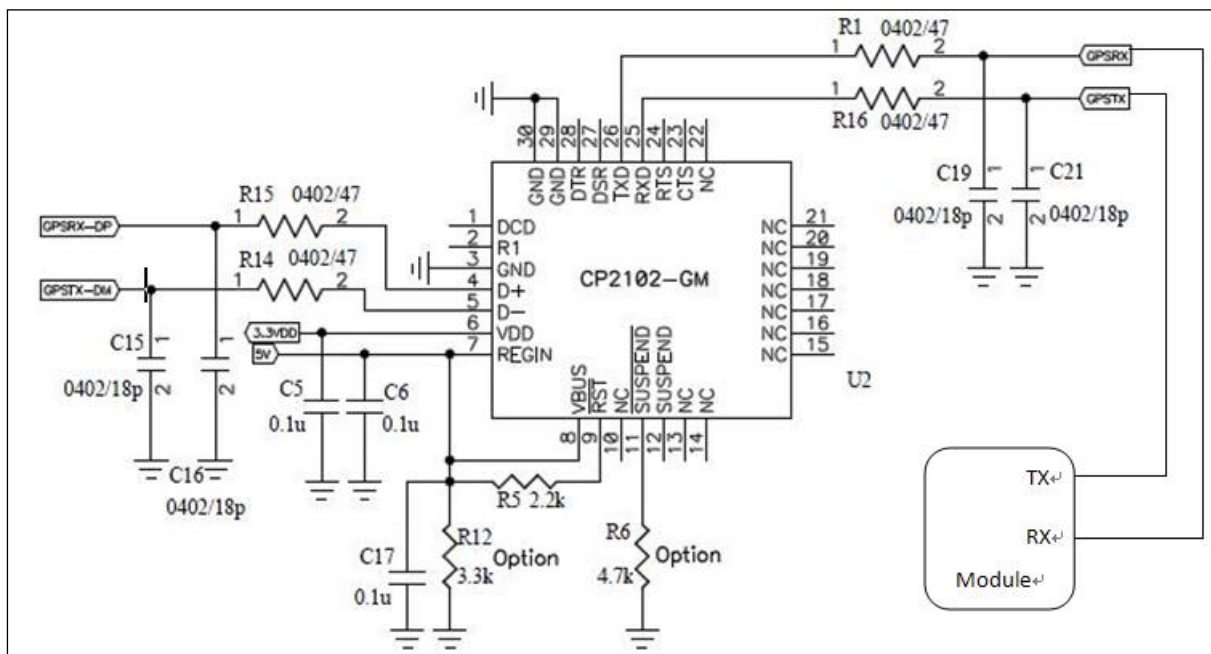


## Appendix IV: UART to USB Interface

If the GlobalTop module you have purchased does not come with USB interface, possibly it is designed to connect the module to an external USB IC. To further enhance the transferring speed, use one that is capable of USB version 2.0. Once the driver for the chosen USB Bridge IC is successfully installed onto Windows or other operating system, the USB Bridge IC will automatically be recognized as a COM port.

Note: proper driver must be installed or else the operating system will not recognize the device!

Please refer to the reference circuits below for the conversion: CP2102 IC is used here as an example.

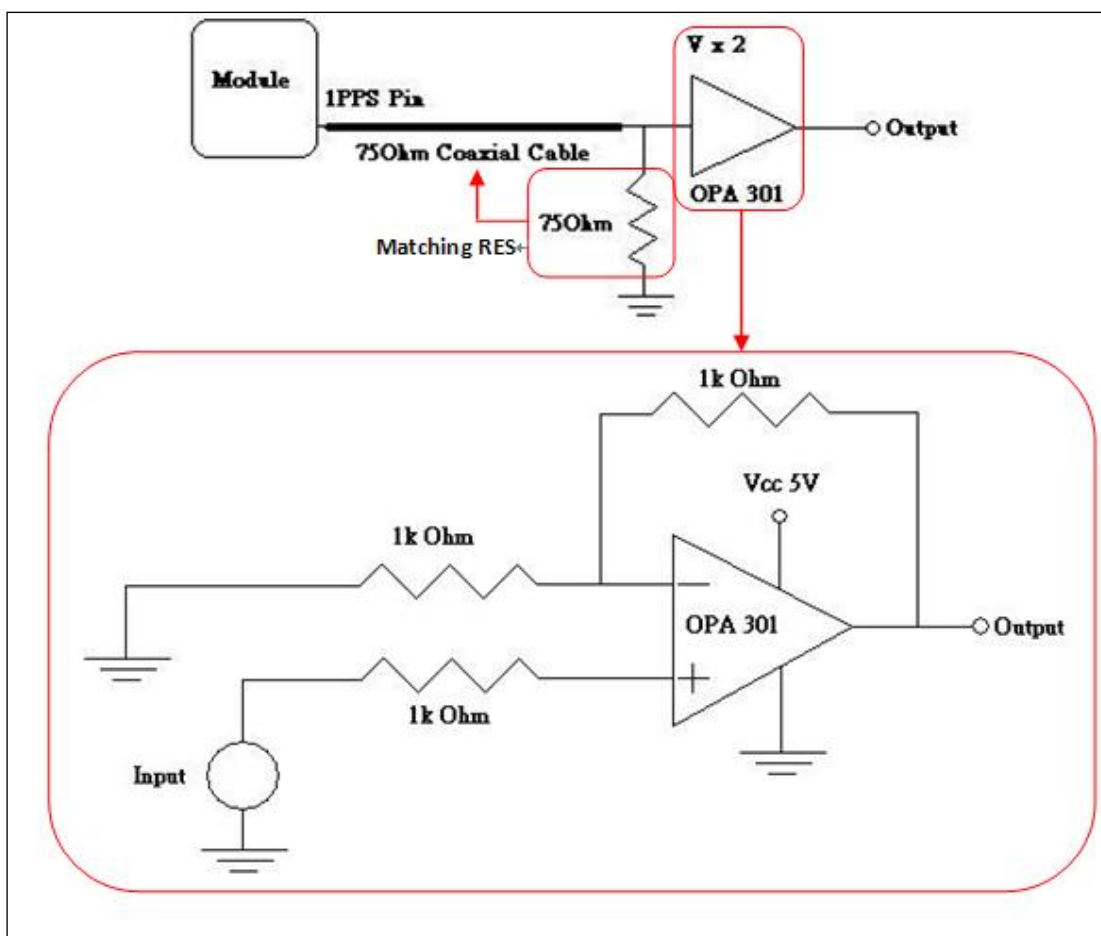


(The Pin29 and Pin30 are the bottom ground pads which are not documented in the IC datasheet. Customer may ignore these two pins in the drawing)

## Appendix V: How to efficiently transfer 1PPS through extended distances

There are several things to note when transferring 1PPS signal through a long cable which will result in 1PPS signal degradation, increase in noise and increase in signal delay. As the 1PPS signal traveling through the communication cable, impedance matching must be implemented to prevent wave form distortion. Depending on user's application, OP-amp can be added to effectively control the accuracy of the voltage level for the 1PPS signal.

When using 1PPS signal for time synchronization, the OP-amp chosen should have a **high slew rate** property to prevent large delay in the 1PPS. The accuracy level of 1PPS for GlobalTop GNSS module is within 100ns. This applies to most of GlobalTop's MTK GNSS modules. The reference design below can be used to improve accuracy to within 80ns.



**1PPS Delay Time:**

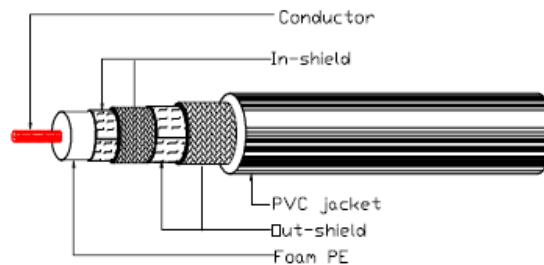
When using coaxial cable for long distances communication using OP-amp for amplification. Please pay close attention to the coaxial cable time delay and the rising time of the wave form due to OP-amp's Slew Rate. A brief explanation for this is given below.

The figure below shows the specification of the coaxial cable with the model number RG6U.

**CONSTRUCTION**

Item		A.Q.L.	Description
Conductor	No./mm	1.02 (± 0.025)	Copper clad steel
	1.02 CCS		
Insulation	No./mm	4.57 (± 0.20)	Foamed PE insulation
	4.57		
Shield	B-APA		Bonded double aluminium foil shield
	64/0.16(AL)	64 (±2); 0.16 (± 0.01)	Aluminium wire braid shield
	APA		Double aluminium foil shield
	48/0.16(AL)	48 (±2); 0.16 (± 0.01)	Aluminium wire braid shield
Jacket	No./mm	7.20 (± 0.30)	Black flame retardant PVC jacket
	7.20 PVC		

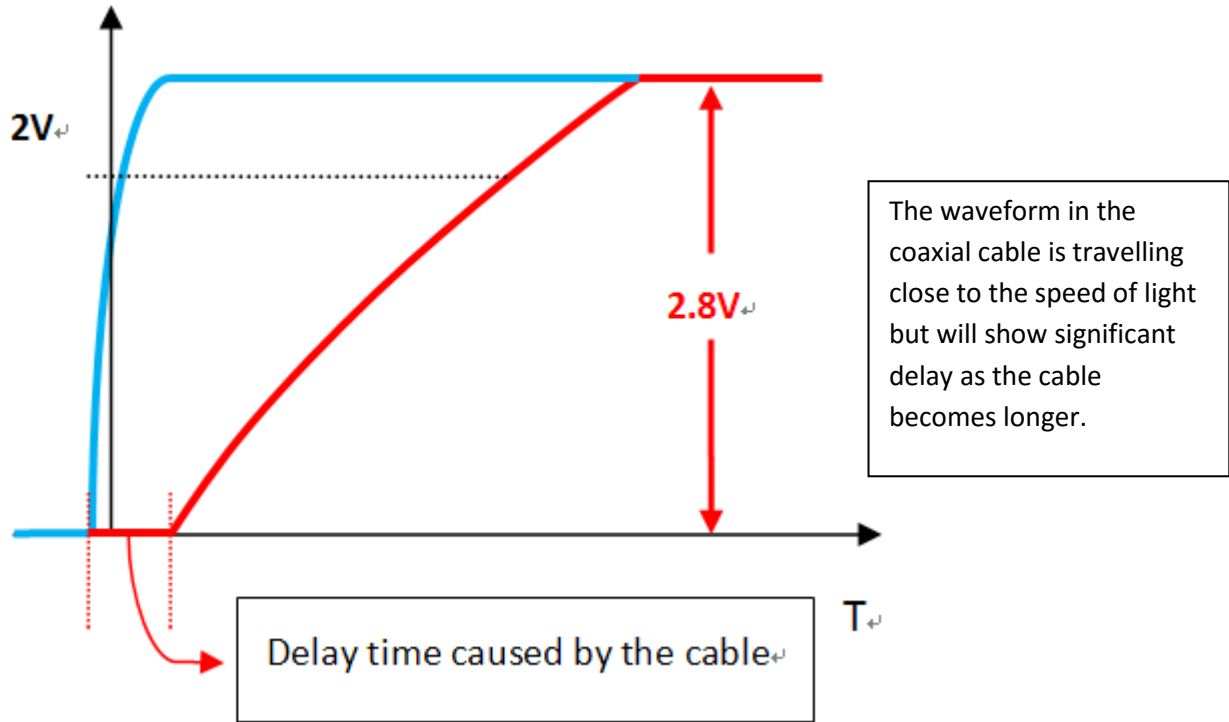
**DESIGN**



**CHARACTERISTICS**

ITEM	Description	Standards	
Conductor Resistance	20°C, conductor DC resistance	<10Ω/100M	
Capacitance		53.2 ± 5pF/M	
Velocity Ratio		83 ± 3%	
Characteristic Impedance	20°C, 200MHz	75Ω ± 3Ω	
Return Loss	20°C, 5 --- 200MHz	>20dB	
Attenuation	20°C -dB/100M (± 10%)	55MHz	4.94
		211MHz	9.43
		400MHz	13.12
		870MHz	19.99
		1000MHz	21.46
		2000MHz	31.82

The delay time cause by the cable:



**Calculating the delay time in respect to the communication cable length**

The simplest method to calculate this is to have a good grasp on the relationships of the three parameters.

1. The speed of the traveling waveform
2. The distance of the wave form traveled
3. The delay time

$$\frac{\ell}{\mu_p} = T$$

$\ell$  = Distance of the wave form traveled (m)

$\mu_p$  = Speed of the waveform (m/s)

T = Delay Time (ns)

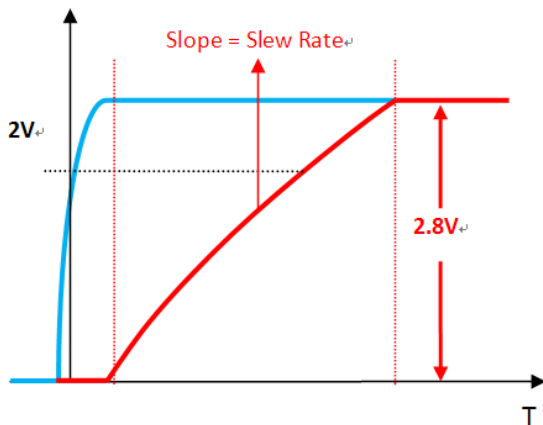
If the speed of the waveform (in coaxial cable) is unknown, it is possible to use a known coaxial cable length to test. For example, the signal travels in 1 meter of coaxial cable can have its delay time measured by the oscilloscope as 5ns and consequently the  $\mu_p$  will be  $2 * 10^8$  (m/s).

If the value of capacitance and inductance for the coaxial cable is known, then  $\mu_p$  (m/s) can also be calculated.

$$\mu_p = \frac{1}{\sqrt{LC}}$$

Moreover, more other coaxial cable electromagnetic fundamental parameters along with the inner and outer diameter of the physical cable can be used in combination to perform theoretical calculations.

**Waveform rising time caused by OP-amp**



The left figure shows Slew Rate: which is the time needed for rising waveform. When the SR (V/μsec) of OPA is higher, the rising time will be shortened as well.

### Recommended OP-amp

If inadequate OP-amp is chosen (i.e. with low slew rate), the delay at the rising time introduced to the waveform can be significantly larger than that of the coaxial wire.

We recommend using **OP-amp 301** to decrease the waveform rising time

Specification of **OP-amp 301**:

- Low Power: 9.5mA (Typ) on 5.5V
- Single Power: 2.7V ~ 5.5V
- High Slew Rate: 80 V/ $\mu$ sec
- Tiny Packages: MSOP and SOT23

### Voltage degradation of communication cable

We have tested a 10m long coaxial cable and only insignificant amount of voltage degradation is observed. Users should not worry this issue.

## Appendix VI: Caution for Reflow Soldering Process

	Details	Suggestions	Notes
1	Before proceeding with the reflow-soldering process, the GNSS module must be pre-baked.	Pre-bake Time: 4 Hours @ 50°to 60°	<b>The maximum tolerated temperature for the tray is 100°C.</b>  After the pre-baking process, please make sure the temperature is sufficiently cooled down to 35°C or below in order to prevent any tray deformation.
2	Because PCBA (along with the patch antenna) is highly endothermic during the reflow-soldering process. Extra care must be paid to the GPS module's solder joint if any signs of cold weld(ing) or false welding.	The parameters of the reflow temperature must be set accordingly to module's reflow-soldering temperature profile.	<b>Double check to see if the surrounding components around the GPS module are displaying symptoms of cold weld(ing) or false welding.</b>
3	Special attentions are needed for PCBA board during reflow-soldering to see if there are any symptoms of bending or deformation to the PCBA board, possibly due to the weight of the module. If so, this will cause concerns at the latter half of the production process.	A loading carrier fixture must be used with PCBA if the reflow soldering process is using rail conveyors for the production.	<b>If there is any bending or deformation to the PCBA board, this might causes the PCBA to collide into one another during the unloading process.</b>
4	Before the PCBA is undergoing a reflow-soldering process, the production operators must check with own eyes to see if there are positional offsets to the module, because it will be difficult to readjust after the module has gone through reflow-soldering process.	The operators must check with own eyes and readjust the position before reflow-soldering process.	<b>If the operator is planning to readjust the module position, please do not touch the patch antenna while the module is hot in order to prevent rotational offset between the patch antenna and module</b>

**Note:** For patch antenna users: patch antenna refers to the patch-on-top antenna which is built on the GPS module (i.e. PA/Gms Module Series) and may not be applicable to all GPS modules.

	Details	Suggestions	Notes
5	Before handling the PCBA, they must be cooled to 35°C or below after they have undergone a reflow-soldering process, in order to prevent positional shift which may occurred when the module is still hot.	<ol style="list-style-type: none"> <li>1. One may use electric fan behind the reflow machine to cool them down.</li> <li>2. Cooling the PCBA can prevent the module from shifting due to fluid effect.</li> </ol>	<b>It is very easy to cause positional offset to the module and its patch antenna when handling the PCBA under high temperature.</b>
6	<ol style="list-style-type: none"> <li>1. When separating the PCBA panel into individual pieces using the V-Cut process, special attentions are needed to ensure that there are sufficient gap between patch antennas.</li> <li>2. If V-Cut process is not available and the pieces must be separated manually, please make sure the operators are not using excess force which may cause rotational offset to the patch antennas.</li> </ol>	<ol style="list-style-type: none"> <li>1. The blade and the patch antenna must have a distance gap greater than 0.6mm.</li> <li>2. Do not use patch antenna as the leverage point when separating the panels by hand.</li> </ol>	<ol style="list-style-type: none"> <li>1. <b>Test must be performed first to determine if V-Cut process is going to be used. Ensure that there is enough space between the blade and patch antenna so that they do not touch one another.</b></li> <li>2. <b>An uneven amount of manual force applied to the separation will likely cause positional shift in patch antenna and module.</b></li> </ol>
7	When separating panel into individual pieces during latter half of the production process, special attentions are needed to ensure the patch antennas do not come in contact with one another in order to prevent chipped corners or positional shifts.	Use tray to separate individual pieces.	<b>It is possible to chip corner and/or cause a shift in position if patch antennas come in contact with each other.</b>

**Note:** For patch antenna users: patch antenna refers to the patch-on-top antenna which is built on the GPS module (i.e. PA/Gms Module Series) and may not be applicable to all GPS modules.

### Other Cautionary Notes on Reflow-Soldering Process:

1. Module must be pre-baked **before** going through SMT solder reflow process.
2. The usage of solder paste must follow “First-in-First-out” principle. Opened solder paste needs to be monitored and recorded in a timely manner (refer to IPQC standards for related documentation and examples)
3. Temperature and humidity must be controlled within SMT production line and storage area. Temperature of 23°C, 60±5% RH humidity is recommended. (please refer to IPQC standards for related documentation and examples)
4. When performing solder paste printing, please notice if the amount of solder paste is excessive or insufficient, as both conditions may lead to defects such as electrical shortage, empty solder ...etc.
5. Make sure the vacuum mouthpiece is able to bear the weight of the GPS module to prevent positional shift during the loading process.
6. Before the PCBA is going through the reflow-soldering process, the operators should check by his/her own eyes to see if there are positional offset to the module.
7. The reflow temperature and its profile data must be measured before the SMT process and match the levels and guidelines set by IPQC.
8. If SMT protection line is running a double-sided process for PCBA, please process GPS module during the second pass only to avoid repeated reflow exposures of the GPS module. Please contact GlobalTop beforehand if you must process GPS module during the 1<sup>st</sup> pass of double-side process.

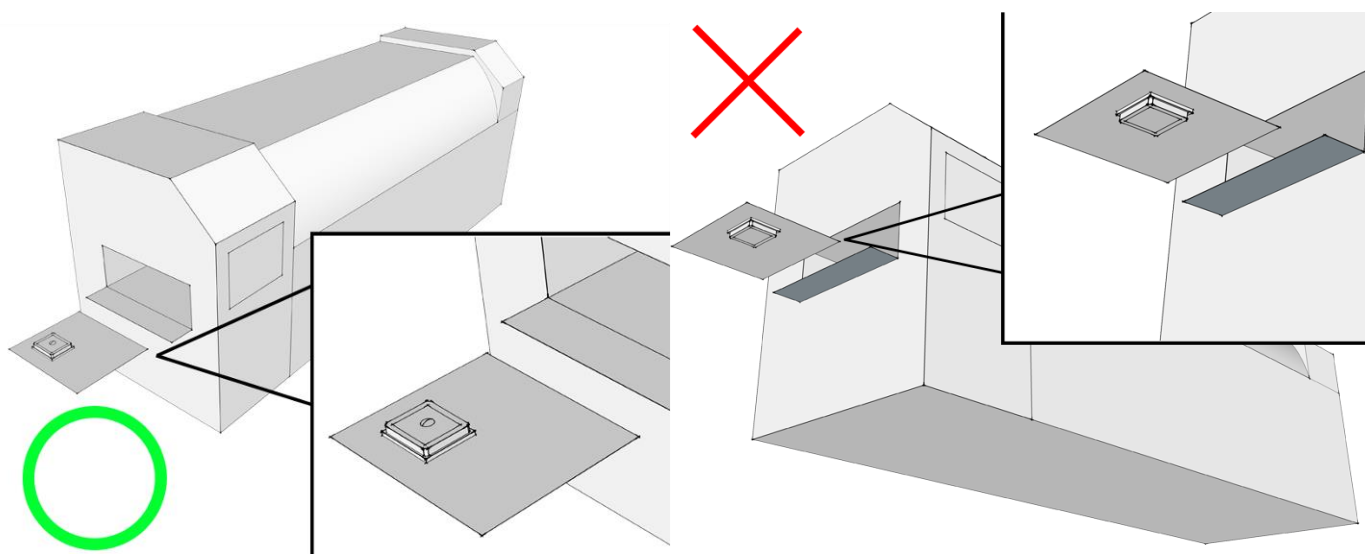


Figure 6.2: Place GPS module right-side up when running reflow-solder process, do not invert.

## **Manual Soldering**

### **Soldering iron:**

Heat Temperature: Under 380°C | Time: Under 3 sec.

### **Notes:**

1. Please do not directly touch the soldering pads on the surface of the PCB board to prevent further oxidation
2. The solder paste must be defrosted to room temperature before use so it can return to its optimal working temperature. The time required for this procedure is unique and dependent on the properties of the solder paste used.
3. The steel plate must be properly assessed before and after use, so its measurement stays strictly within the specification set by SOP.
4. Please watch out for the spacing between soldering joint, as excess solder may cause electrical shortage
5. Please exercise with caution and do not use extensive amount of flux due to possible siphon effects on neighboring components, which may lead to electrical shortage.
6. Please do not use the heat gun for long periods of time when removing the shielding or inner components of the GPS module, as it is very likely to cause a shift to the inner components and will leads to electrical shortage.